Chapter 9

Thermal Analysis

Finite Element Modeling and Simulation with ANSYS Workbench

Introduction

- Many engineering problems are thermal problems in nature. Devices such as appliances, electronics, engines, and heating, ventilation and air conditioning systems need to be evaluated for their thermal performance.

- In thermal analysis, the resulting temperature and heat flux distributions and structural response under thermal loading are important knowledge in assuring design success of thermal engineering products.

KENT STATE UNIVERSITY

Review of Basic Equations

Thermal and thermal stress analyses are briefly reviewed …

Review of Basic Equations

- ***Steady-state thermal analysis***

  Find the temperature or heat flux distributions in structures when a thermal equilibrium is reached.

- ***Transient thermal analysis***

  Determine the time history of how the temperature profile and other thermal quantities change with time.

- ***Thermal stress analysis***

  Examine thermal expansion or contraction of engineering materials that leads to thermal stress in structures.

Review of Basic Equations

Thermal Analysis

For temperature field in a 1-D space, such as a bar, we have the following *Fourier heat conduction equation*

$$f_x = -k\frac{\partial T}{\partial x}$$

where

f_x = heat flux per unit area
k = thermal conductivity
$T = T(x, t)$ = temperature field

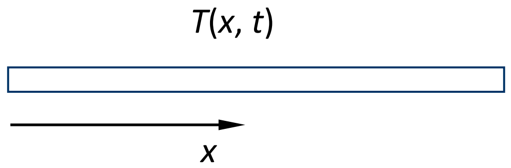


Figure 9.1. The temperature field $T(x, t)$ in a 1-D bar model.

For 3-D case, we have

$$\begin{Bmatrix} f_x \\ f_y \\ f_z \end{Bmatrix} = -\mathbf{K}\begin{Bmatrix} \partial T/\partial x \\ \partial T/\partial y \\ \partial T/\partial z \end{Bmatrix}$$

where

f_x, f_y, f_z = heat flu in the *x, y* and *z* direction, respectively.

For isotropic materials, the conductivity matrix

$$\mathbf{K} = \begin{bmatrix} k & 0 & 0 \\ 0 & k & 0 \\ 0 & 0 & k \end{bmatrix}$$

Review of Basic Equations

The *equation of heat flow* is given by

$$-\left[\frac{\partial f_x}{\partial x}+\frac{\partial f_y}{\partial y}+\frac{\partial f_z}{\partial z}\right]+q_v=c\rho\frac{\partial T}{\partial t}$$

where
q_v = rate of internal heat generation per unit volume
c = specific heat
ρ = mass density

For steady state case ($\frac{\partial T}{\partial t}=0$) and isotropic materials, we can obtain

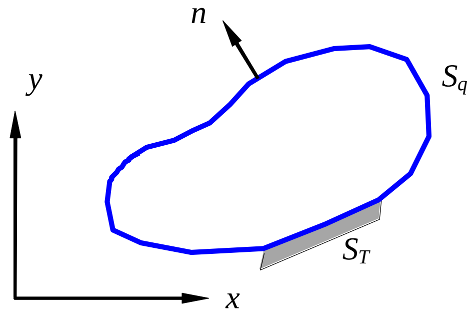
$$k\nabla^2 T=-q_v$$

This is a Poisson equation, which can be solved under given boundary conditions.

Boundary conditions for steady state heat conduction problems are

$T=\overline{T},$ on S_T ;

$Q\equiv -k\frac{\partial T}{\partial n}=\overline{Q},$ on S_q ;



Note at any point on $S=S_T\cup S_q$ only one type of BCs can be specified.

Review of Basic Equations

Finite Element Formulation for Heat Conduction

For heat conduction problems, we can establish the following FE equation

$$\mathbf{K}_T \mathbf{T} = \mathbf{q} \qquad \text{Eq. (1)}$$

where

- $\mathbf{K}_T$ = conductivity matrix
- $\mathbf{T}$ = vector of nodal temperature
- $\mathbf{q}$ = vector of thermal loads

The element conductivity matrix is

$$\mathbf{k}_T = \int_V \mathbf{B}^T \mathbf{K} \mathbf{B} \, dV$$

This is obtained in a similar way as for the structural analysis, i.e., by starting with the interpolation $T = \mathbf{N}\mathbf{T}_e$

For transient (unsteady state) heat conduction problems, we have

$$\frac{\partial T}{\partial t} \neq 0$$

In this case, we need to apply finite difference schemes (use time steps and integrate in time), as in the transient structural analysis, to obtain the transient temperature fields.

Review of Basic Equations

Thermal Stress Analysis

To determine the thermal stresses due to temperature changes in structures, we can

- Solve Eq. (1) first to obtain the temperature (change) fields.
- Apply the temperature change ΔT as initial strains (or initial stresses) to the structure to compute the thermal stresses due to the temperature change.

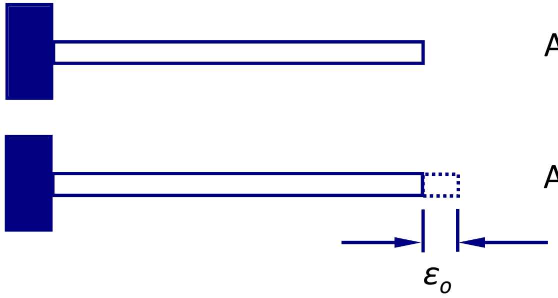


Figure 9.3. Expansion of a bar due to increase in temperature.

Review of Basic Equations

1-D Case

To understand the stress-strain relations in cases of solids undergo temperature changes, we first examine the 1-D case. We have for the thermal strain (or initial strain)

$$\varepsilon_o = \alpha \Delta T$$

in which,

α: the coefficient of thermal expansion

$\Delta T = T_2 - T_1$: change of temperature

Total strain is given by:

$$\varepsilon = \varepsilon_e + \varepsilon_o$$

where

ε_e is the elastic strain due to mechanical load.

Thus the total strain can be written as

$$\varepsilon = E^{-1}\sigma + \alpha \Delta T$$

Or, inversely, the stress is given by

$$\sigma = E(\varepsilon - \varepsilon_o)$$

An Example in FEA: Spring System

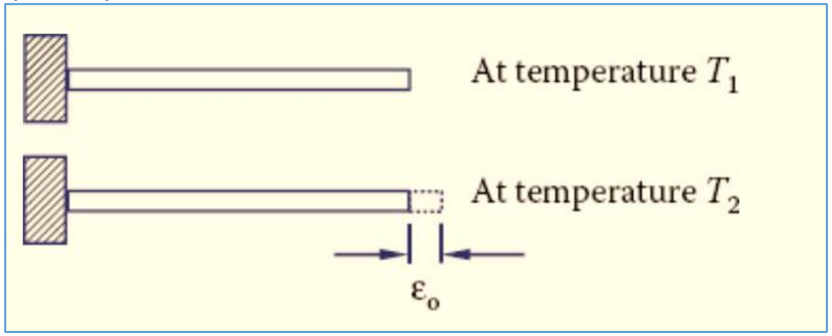
Example 9.1

Consider the bar under thermal load ΔT as shown in Figure 9.3.

(a) If no constraint on the right-hand side, i.e., the bar is free to expand to the right, then we have

$$\varepsilon = \varepsilon_o, \ \varepsilon_e = 0, \ \sigma = 0$$

That is, there is no thermal stress in this case.



(b) If there is a constraint on the right-hand side, that is, the bar cannot expand to the right, then we have

$$\varepsilon = 0, \quad \varepsilon_e = -\varepsilon_o = -\alpha \Delta T, \qquad \sigma = -E\alpha\Delta T$$

Thus, thermal stress exists.

From this simple example, we see that the way in which the structure is constrained has a critical role in inducing the thermal stresses.

Review of Basic Equations

2-D Case

For plane stress, we have

$$\boldsymbol{\varepsilon}_o = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix}_o = \begin{Bmatrix} \alpha \Delta T \\ \alpha \Delta T \\ 0 \end{Bmatrix}$$

For plane strain, we have

$$\boldsymbol{\varepsilon}_o = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix}_o = \begin{Bmatrix} (1+\nu)\alpha \Delta T \\ (1+\nu)\alpha \Delta T \\ 0 \end{Bmatrix}$$

in which, ν is the Poisson's ratio.

Review of Basic Equations

3-D Case

$$\boldsymbol{\varepsilon}_o = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix}_o = \begin{Bmatrix} \alpha\Delta T \\ \alpha\Delta T \\ \alpha\Delta T \\ 0 \\ 0 \\ 0 \end{Bmatrix}$$

Observation: Temperature changes do not yield shear strains.

In both 2-D and 3-D cases, the total strain can be given by the following vector equation

$$\boldsymbol{\varepsilon} = \boldsymbol{\varepsilon}_e + \boldsymbol{\varepsilon}_o$$

And the stress-strain relation is given by

$$\boldsymbol{\sigma} = \mathbf{E}\boldsymbol{\varepsilon}_e = \mathbf{E}(\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}_o)$$

Review of Basic Equations

Notes on FEA for Thermal Stress Analysis

- Need to specify α for the structure and ΔT on the related elements (which experience the temperature change).
- Note that for linear thermoelasticity, same temperature change will yield same stresses, even if the structure is at two different temperature levels.
- Differences in the temperatures during the manufacturing and working environment are the main cause of thermal (residual) stresses.

Examples of thermal and thermal stress analyses are illustrated …

Modeling of Thermal Problems

- Heat transfers in three ways in our environment
  - Conduction

    modeled by solving the resulting heat balance equations for the nodal temperatures under specified thermal boundary conditions.

  - Convection

    modeled as a surface load with a user-specified heat transfer coefficient and a given bulk temperature of the surrounding fluid.

  - Radiation

    modeled by using the radiation link elements or surface effect elements with the radiation option.

Modeling of Thermal Problems

- Steady-state thermal analysis: Need thermal conductivity as the material input.
- Transient thermal analysis: Material properties such as density, thermal conductivity and specific heat are needed as input parameters.
- Thermal stress analysis: Material input parameters include Young's modulus, Poisson's ratio and thermal expansion coefficient.

Modeling of Thermal Problems

❑ Thermal Analysis

A heat sink is a device commonly used to dissipate heat from a CPU in a computer. In this heat sink model, a given temperature field ($T = 120$) is specified on the bottom surface and a heat flux condition ($Q \equiv -k\frac{\partial T}{\partial n} = -0.2$) is specified on all the other surfaces.

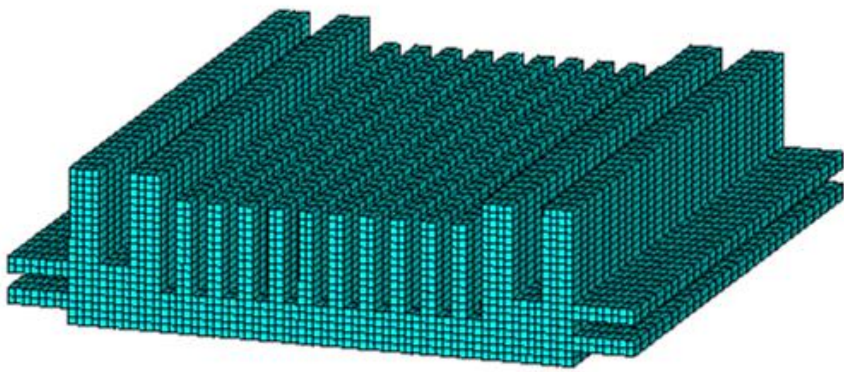


Figure 9.4. A heat-sink model used for heat conduction analysis.

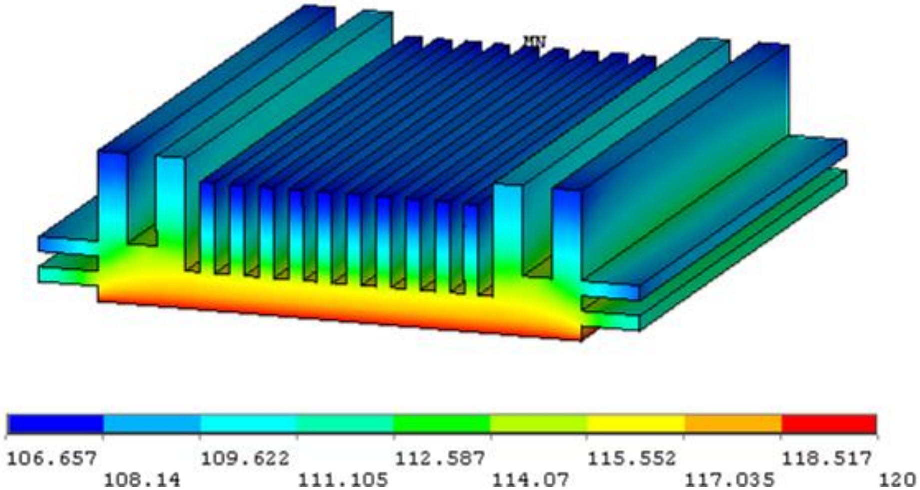


Figure 9.5.Computed temperature distribution in the heat-sink.

Modeling of Thermal Problems

❑ Thermal Stress Analysis

Next, we study thermal stresses in structures due to temperature changes.

We assume that the plate is made of steel with the Young's modulus E = 200 GPa, Poisson's ratio ν = 0.3 and thermal expansion coefficient α = 12x10^{-6} 1/°C. The plate is applied with a uniform temperature increase of 100 °C.

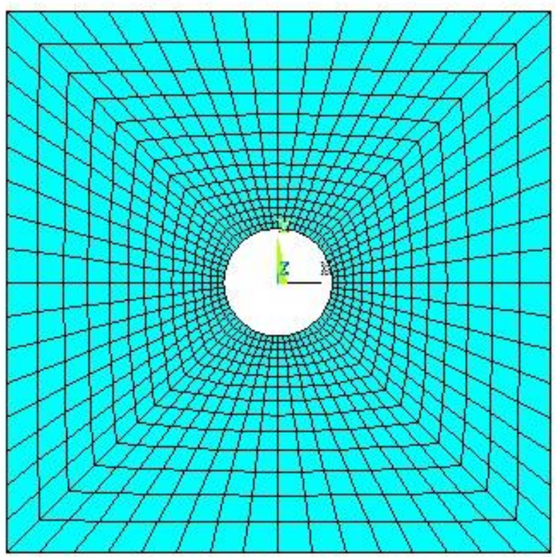


Figure 9.6. A square plate with a center hole and under a uniform temperature load.

Modeling of Thermal Problems

The computed thermal stresses in the plate under two different types of constraints.

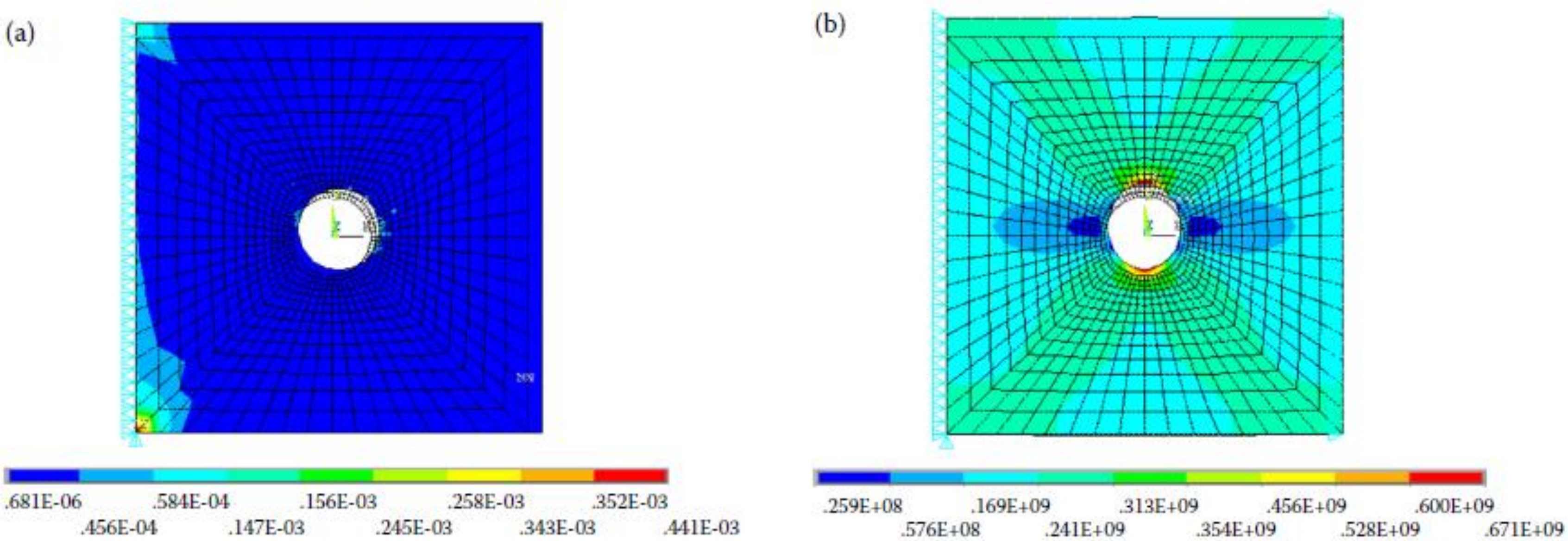


Figure 9.7. Thermal (von Mises) stresses in the plate: (a) When the plate is constrained at left side only (thermal stresses = 0); (b) When the plate is constrained at both left and right sides.

Axisymmetric analysis of a heat sink…

Case Study with ANSYS Workbench

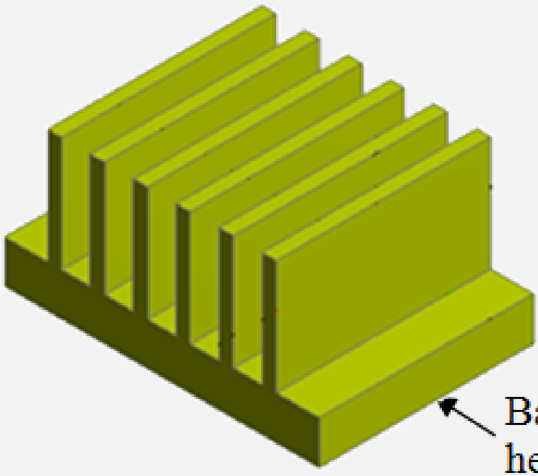
<Problem Description> Heat sinks are commonly used to enhance heat dissipation from electronic devices. In this Case Study, we conduct thermal analysis of a heat sink made of aluminum with thermal conductivity k = 170 W/(m·K), density ρ = 2800 kg/m^3, specific heat c = 870 J/(Kg·K), Young's modulus E = 70 GPa, Poisson's ratio v = 0.3 and thermal expansion coefficient α = 22x10^{-6}/°C. A fan forces air over all surfaces of the heat sink except for the base, where a heat flux q' is prescribed. The surrounding air is 28°C with a heat transfer coefficient of h = 30W/(m^2·°C).

(Part A): Study the steady-state thermal response of the heat sink with an initial temperature of 28°C and a constant heat flux input of q' = 1000W/m^2.

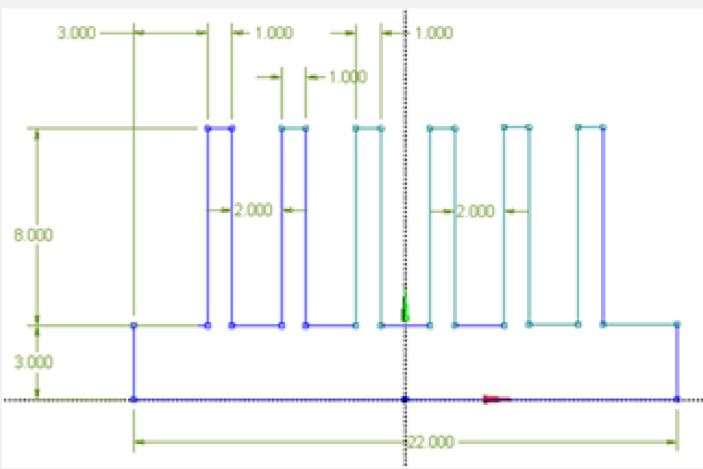
(Part B): Suppose the heat flux is a square wave function with period of 90 seconds and magnitudes transitioning between 0 and 1000W/m^2. Study the transient thermal response of the heat sink in 180 seconds by using the steady-state solution as the initial condition.

(Part C): Suppose the base of the heat sink is fixed. Study the thermal stress response of the heat sink by using the steady-state solution as the temperature load.

Case Study with ANSYS Workbench



Base of the heat sink



All dimensions are in millimeters.

Material: Aluminum

k= 170 W/(m·K)

ρ= 2800 kg/m^3; c = 870 J/(kg·K)

E = 70 GPa; ν = 0.3

α = 22x10^{-6} 1/°C

Boundary Conditions:

Air temperature of 28°C; h = 30W/(m^2·°C).

Steady-state: q' = 1000W/m^2 on the base.

Transient: Square wave heat flux on the base.

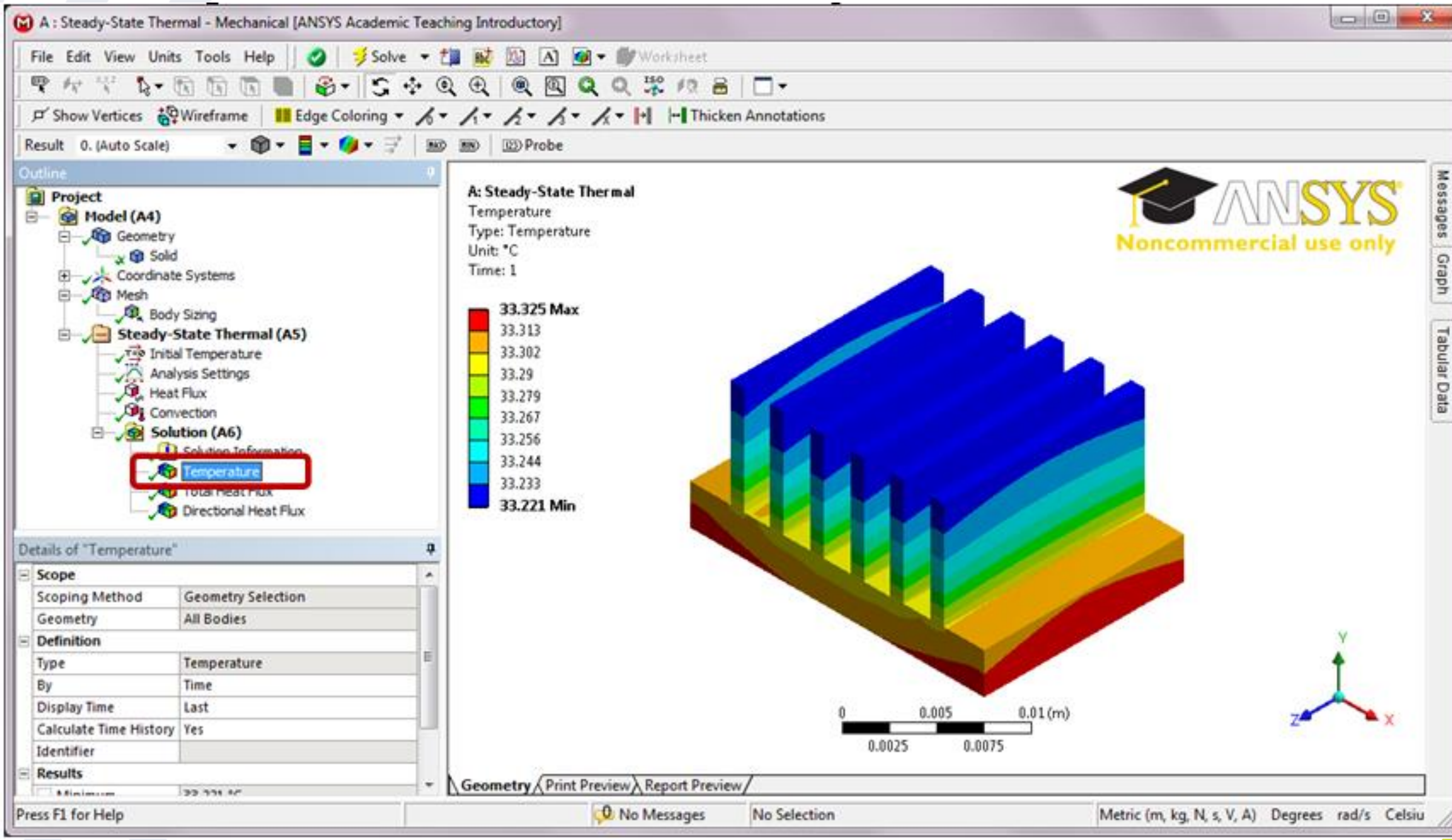
Initial Conditions:

Steady-state: Uniform temperature of 28°C.

Transient: Steady-state temperature results.

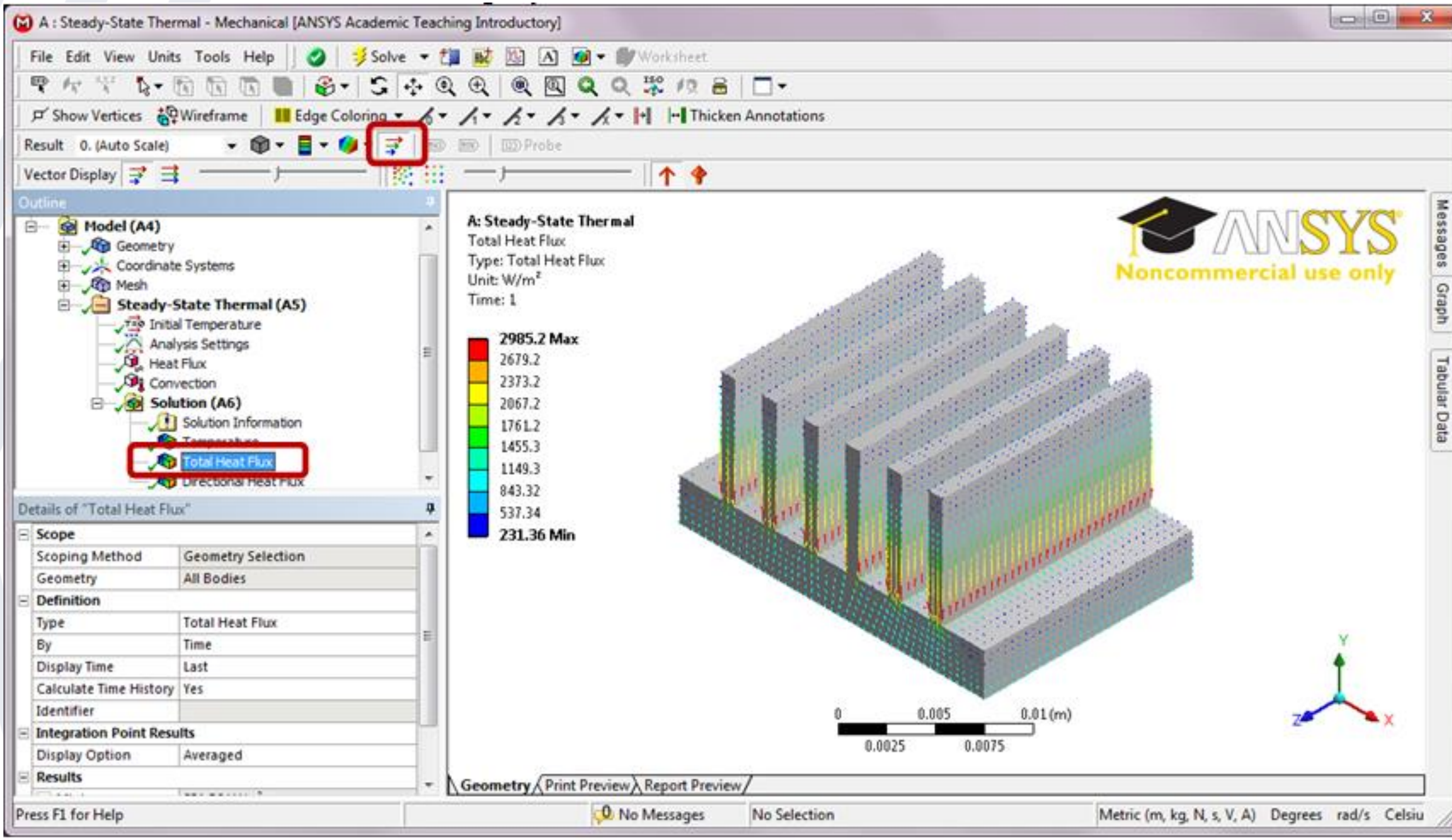
Case Study with ANSYS Workbench

Run a ***Steady-State Thermal Analysis*** to view the temperature results.



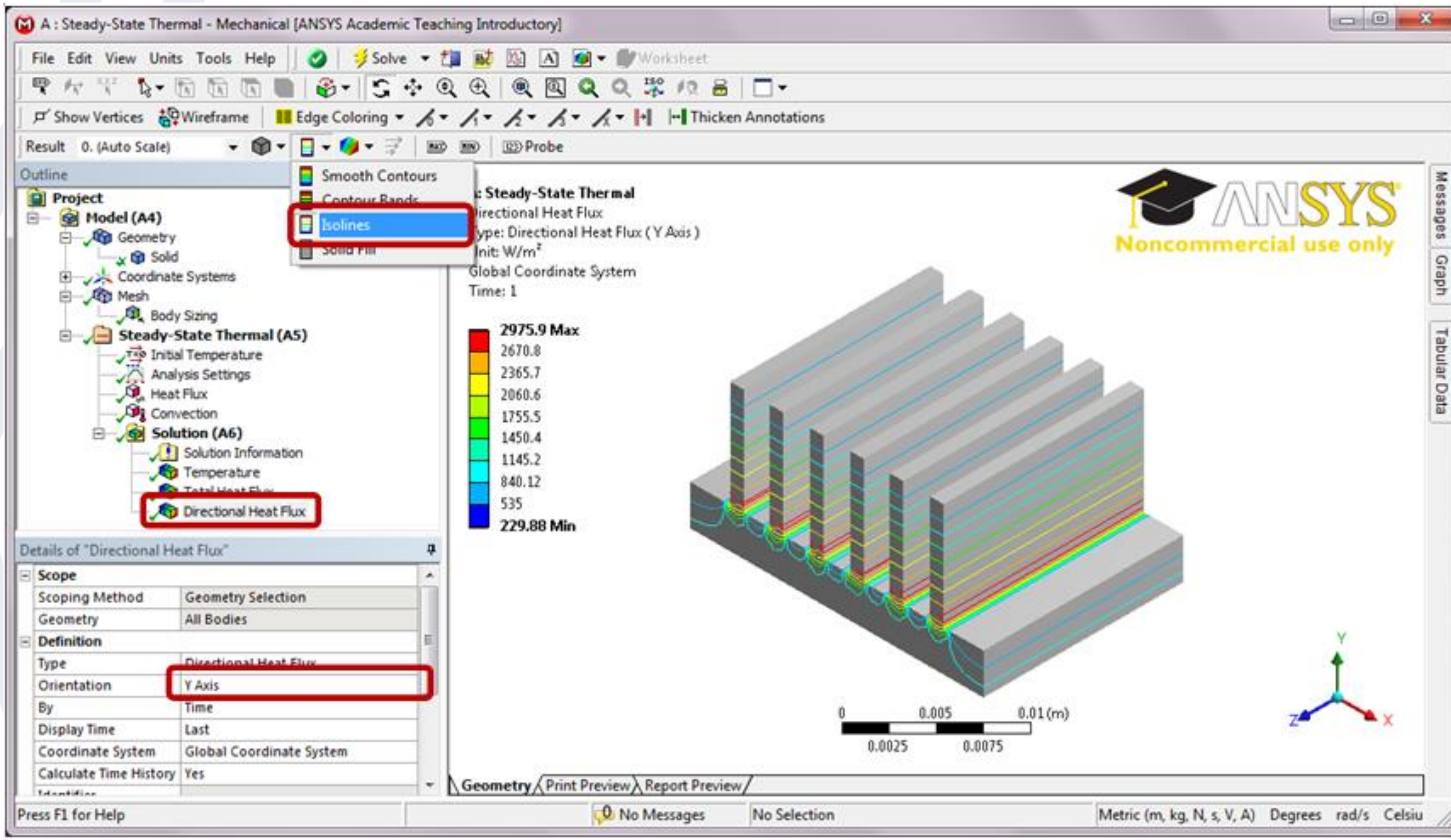
Case Study with ANSYS Workbench

Select ***Total Heat Flux*** to display the heat flux with directional arrows.



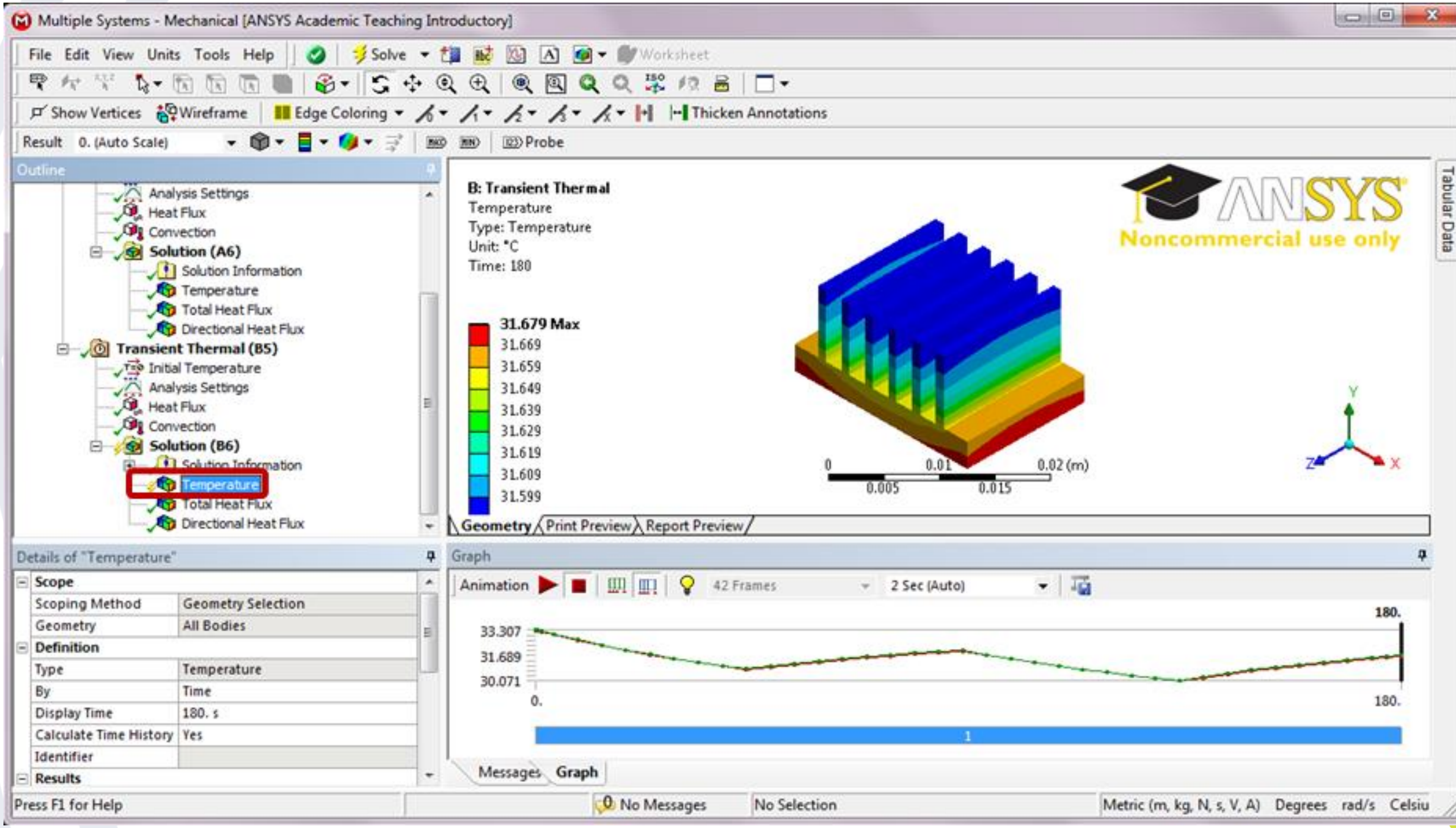
Case Study with ANSYS Workbench

Select ***Directional Heat Flux*** to review the heat flux isolines along ***Y-axis***.



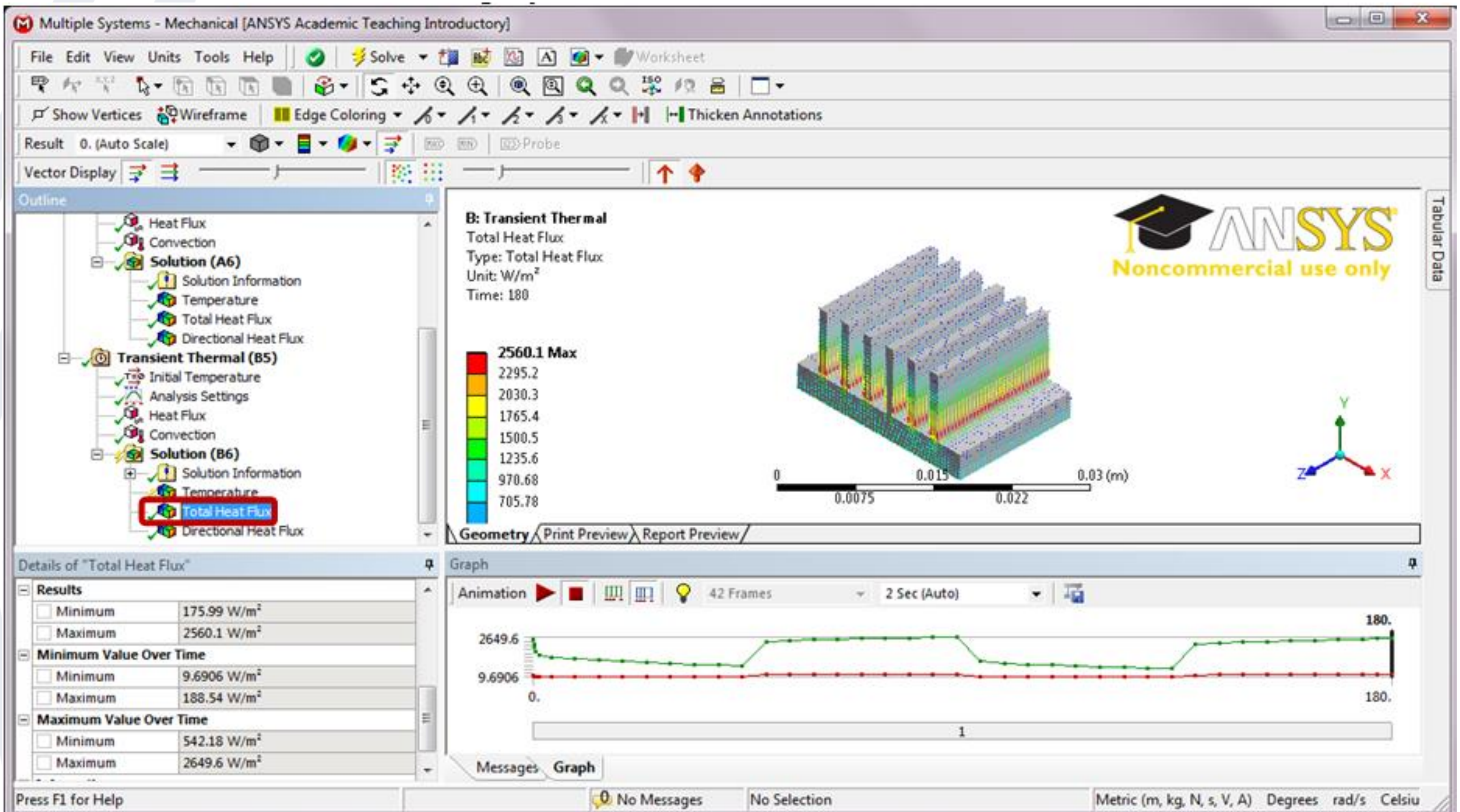
Case Study with ANSYS Workbench

Run a ***Transient Thermal Analysis*** to review the transient thermal results.



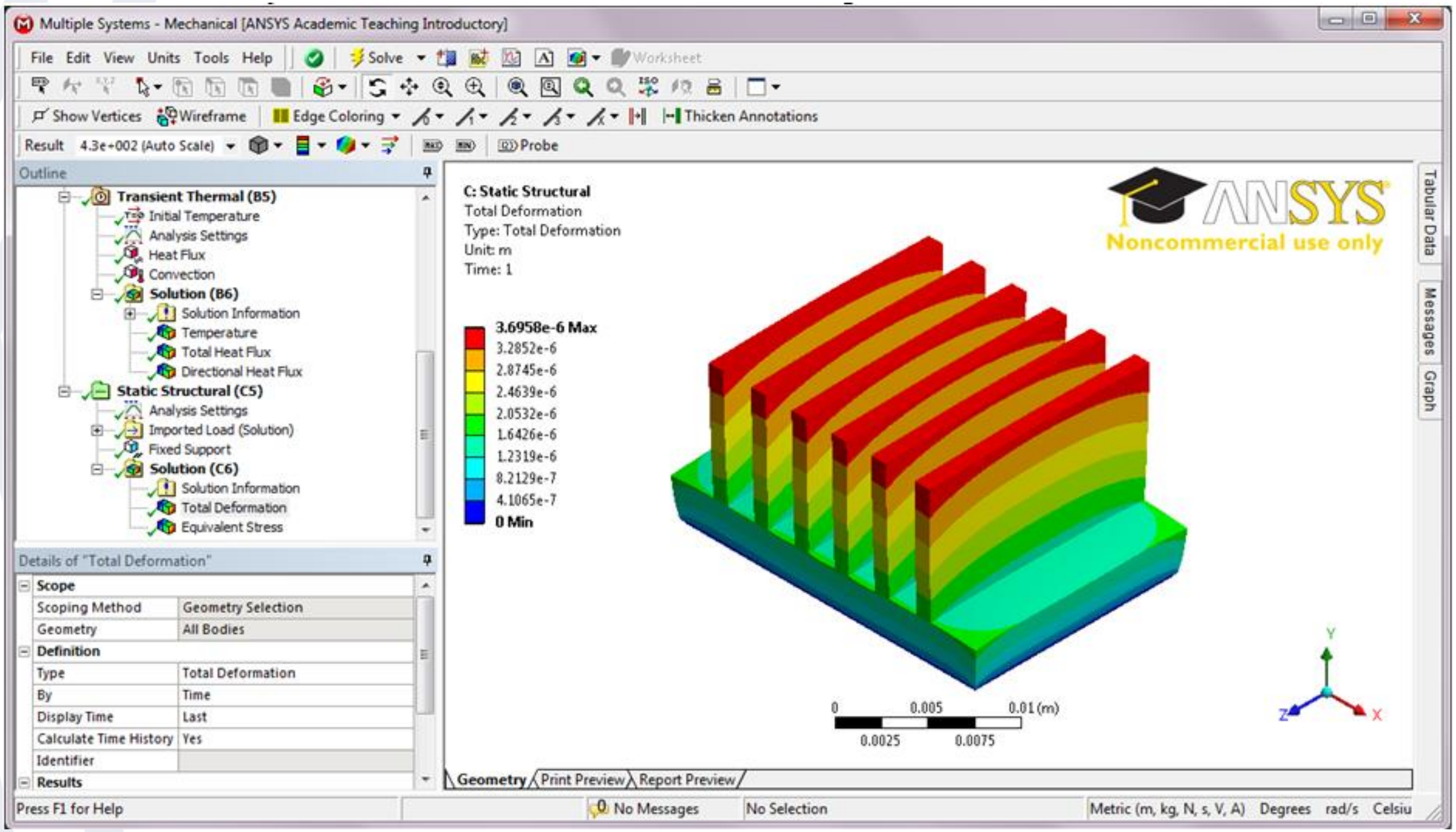
Case Study with ANSYS Workbench

Select ***Total Heat Flux*** to display the transient heat flux in the heat sink.



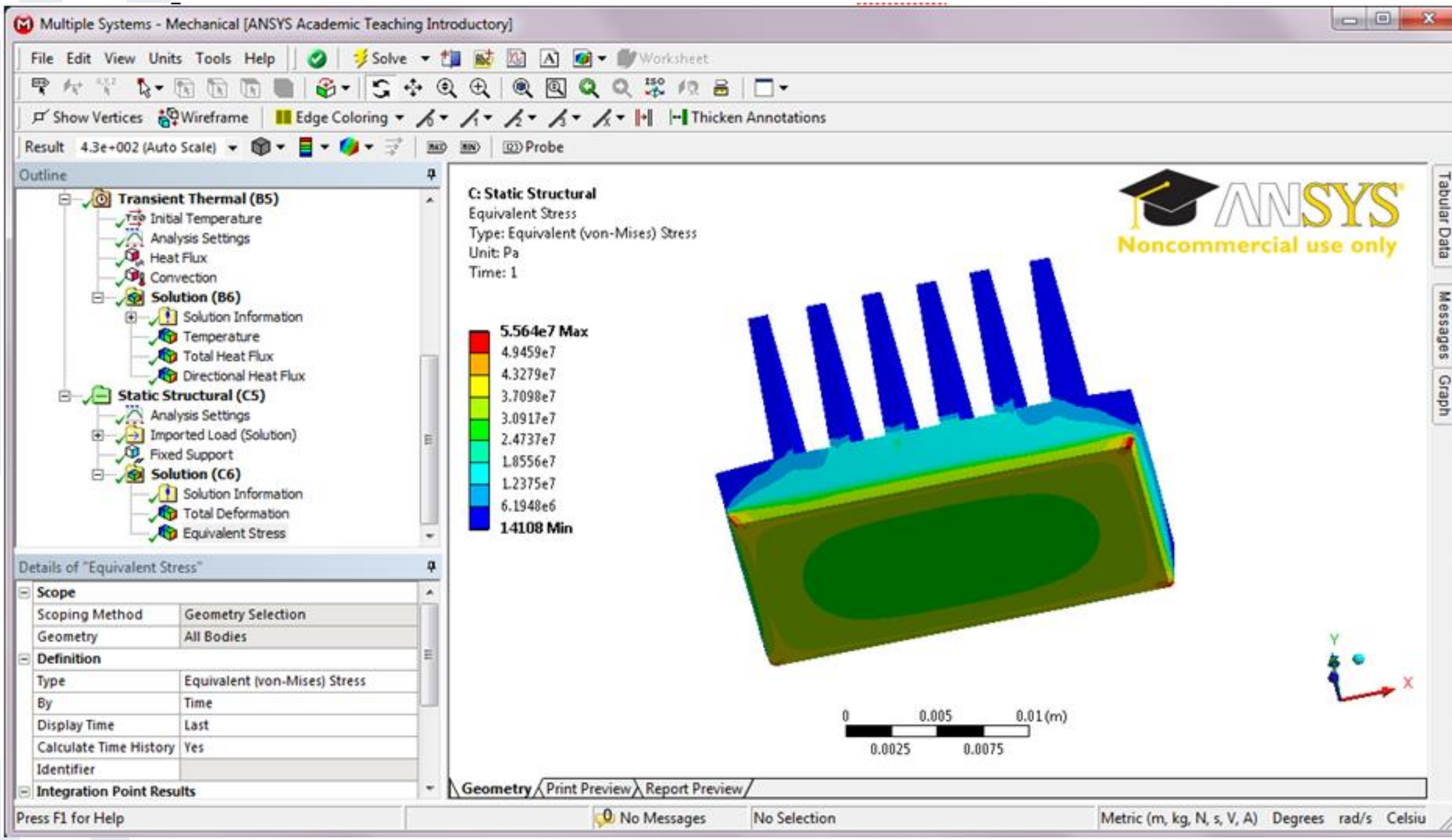
Case Study with ANSYS Workbench

Run a ***Thermal Stress Analysis*** to review the temperature-induced deformation.



Case Study with ANSYS Workbench

Select ***Equivalent Stress*** in the ***Outline*** to review von Mises stress results.



Summary

In this chapter,

- The governing equations for heat conduction problems and the FEA formulation are discussed.
- Thermal stresses due to changes of temperatures in structures are discussed.
- The effects of constraints of the structures on the thermal stresses are emphasized.

Thermal Analysis Tutorial-steady state Conduction

https://www.youtube.com/watch?v=HvViM6vexyw&t=27s

Thermal Analysis Tutorial-Combined thermal and stress transient analysis

https://www.youtube.com/watch?v=KhVXU1WrZro&t=1267s

Coupled Analysis (Structural + Thermal) using ANSYS Workbench-Steady State

https://www.youtube.com/watch?v=KGWk9bcWncY